ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology
Vol. 2, Issue 6, June 2015

CFD Simulation of a Co–Current Spray Dryer for Silica Powder Production

Rajashekhara M C[1], Raghavendra N[2]

P.G Student, Mechanical Department, The oxford college of Engineering, Bangalore , India[1]

Asst., Professor, Department of Mechanical Engineering, The oxford college of Engineering, Bangalore, India[2]

Abstract: This paper presents the prediction of air flow, temperature patterns and mass fraction of water vapour in a co current pilot plant spray dryer fitted with pressure swirl nozzle using 3 dimensional model. The modeling was done with CFD package star ccm+ 9.04, in which the droplet/particle phase is modeled with Langrangian approach and hot air stream is modeled with Euler approach, K-ε model is used to analysis the turbulence swirl in the spray dryer.
Good results were obtained with the modified balanced double tangential air inlets spray dryer when compared with the conventional single tangential air inlet spray dryer . the problem of stick of the wet particles on the walls of the spray dryer and cone part of the spray dryer was eliminated in the modified model. A higher uniformity of temperature and air mass fraction on the horizontal planes of the spray dryer were obtained by the modified design of the spray dryer.
This work was carried with multiphase flow and the feed considered for the work is silica slurry (40% Si, 60%liquid water) for the analysis.In this work the effects of the air flow pattern on the droplets trajectory, residence time, distribution of droplets, and the deposition of droplets on the wall were compared with the conventional model and the modified air inlet design model.

Keywords: Spray drying; Wall deposition; turbulence; Computational fluid dynamics (CFD)

I. INTRODUCTION

Spray dryer is an essential plant for the manufacturing of many products with specific properties of powder example; food products, chemical, pharmaceuticals, ceramics etc., in spite of the wide use of spray dryers, they are still designed mainly on the basis of experience and pilot experiments [1].

Due to the lack of information about the inside operating parameters. One big problem facing by spray dryer designer and operator is the complexity of the spray and air mixing problem in the spray dryer chamber [2].where the air flow pattern exists inside the spray dryer chamber is considered as one of the primary factor that controls the residence time of the particles to be dry, which intern controls the out put product of the spray dryer particle size and its specific properties. For the higher heat sensitive product like silica or milk, the residence time (the time elapsed by the particle in the hot air stream) is very important parameter to analysis but this only can not affect to the particle degradation because in the spray dryer there may be occurrence of very hot and cold zones, which are the main reasons for the particle degradation and wet particle stick of the wall. This makes the unequal drying of the particles by the air stream in the spray dryer chamber [3].

The main spray dryer operability phenomenon is the wall deposition of wet particles affected by the air flow velocity pattern in the spray dryer as well as its temperature, when the wet particle droplet contact the spray dryer wall which sticks to there, if the adhesive force of attraction of wall and water is sufficient to stick, when the water content in the particle is more the stickiness is more. These depositions are dangerous because they can cause damage to the chamber wall or they can char resulting in a potential explosions hazards[4].Oakley et al.[5], kieviet[6]and others applied the techniques of computational fluid dynamics to analysis the spray dryer successfully. All most all of these earlier works assumes flow as in the dryer are two dimensional and axi-symmetric. In order to reduce the demand on computational resources. There is a need of 3D nature of the flows for the clear demonstrate of experimental evidences [7]. Even though the 3D model nature of the flows are done there is a need of optimization of the air swirl to increase residence time as well as good turbulence to proper mixing of air stream and particles. There again a need of multiphase flow analysis for the slurry and air flow pattern analysis that is lagrange-euler models of analysis. Here we need to know that what happening in the operation of the spray dryer, how the air gets deflected in the spray dryer entry level how the interaction of the particles and air stream happening. So we need to analyze the spray dryer chamber from the beginning of the air inlet because the entry of air stream into the spray dryer is the main point where the swirl can be created naturally by the tangential inlet. So many people worked on the pure liquid droplets evaporation In the modeling of atomization of fuels in engines, but this work is similar to that and some complex of the multiphase flow it diverts the analysis from that simple evaporation process.

II. METHODOLOGY

Modeling approach

Turbulent multiphase flow is the general flow in spray dryer. In this two main methods of approach are there 1.euler and 2. Lagrange, in which since the gas or hot air flow is continuous fluid flow and the sprayed particles flow is discontinuous flow hence Euler approach is used to hot air flow and Lagrange approach is used to particulate

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
Vol. 2, Issue 6, June 2015

phase flow is more suitable to study. Hence the same approach is used for this study before doing Lagrange approaches here need to do Euler approach of hot air stream. This was done by solving the navier stokes and continuity equations on a control volume of grid. Subsequently the particles trajectory are tracked individually by solving the mass, energy and momentum exchange equations these characteristic transfer terms are added to the source term of N-S equation for the calculation of air stream flow.

By the scheme of Particle-In-Cell model[1],the repeated calculation of air stream flow and particle tracking is done until the solution is converged with consideration source term. By solving the droplet motion equation and its heat and mass transfer it is calculated that the particle trajectory its velocity, size of droplet, temperature and other history are calculated [15, 16]. The simple drying kinetics for the evaporation of moisture from the droplet is

$$\frac{dm}{dt} = k_e A_d M_w [C_{ws} - C_{w\infty}] \quad \dots\dots\dots\dots\dots\dots\dots \quad 1$$

This is mainly based on the Fick's law of mass transfer, in which the diffusion gradient of droplet moisture flux into the hot air phase with related to the moisture concentration gradient between the particle surface and the hot air stream.

Assumed that the saturated vapour pressure P_{sat} at the interphase. So that the concentration of moisture at the droplet surface is calculated by

$$C_{ws} = \frac{P_{sat}}{RT_d} \quad \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots\dots \quad 2$$

And the moisture concentration in the hot air stream is evaluated by

$$C_{w\infty} = X_w \frac{P}{RT_\infty} \quad \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots \quad 3$$

Where,

X_w = the local bulk mole fraction of water vapour

P = is the local absolute pressure

T_∞ = the local bulk temperature in the hot air stream...

The mass transfer co efficient in the equation (1) is calculated from the showed number with appropriate co relation

$$Sh_{AB} = \frac{k_c d_d}{D_m} = 2.0 + 6.0 Re_d^{0.5} Sc^{0.33} \quad \dots\dots\dots\dots\dots \quad 4$$

A turbulence model of K-ε model is used to analyze the swirl (turbulence) in the air stream and particle path.

III. CASE STUDY

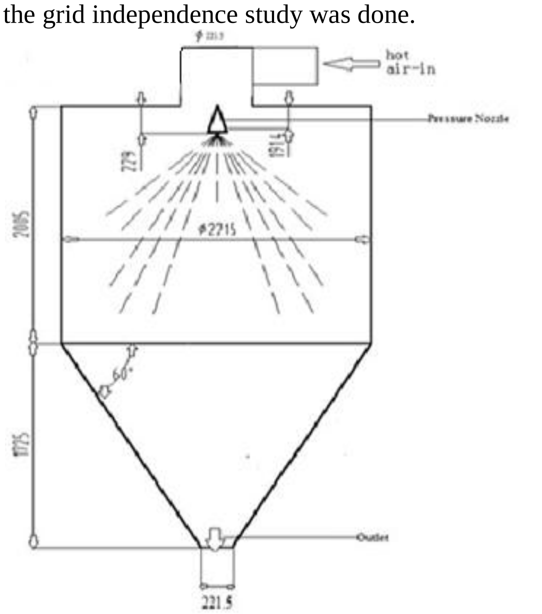
For the CFD simulation of the spray dryer here used is a co current pilot plant spray dryer. The geometry and the air size are as per the standard dimension of industrial spray dryer.

But in this work the comparison was made between conventional single tangential air inlet spray dryer and the modified balanced double tangential air inlet spray dryer. In both the case tangential inlets are considered to create natural swirl from which residence time to increase

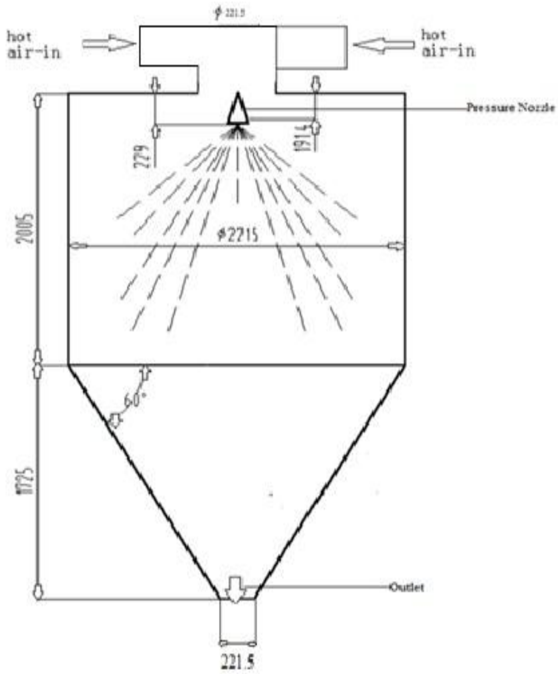
The nozzle is kept in –z direction and the air inlets are in the ±x-axis because this is the most important to consider the full air inlet divergences for the analysis of the distribution of air inside the spray dryer.

The outlet for the spray dryer is common for the air + moisture and particles because these are separated in the cyclone separator or in the bag house filter(which are not included to the study).

For the modeling and simulation purpose the CFD package star ccm+ 9.04 is used. The geometry used for the study is as shown in figure 1 and the meshing done to it with the size of 0.03 m thus the total number of cells generated are 583144 cells with face validity=1.25 ,volume ratio =0.63 e-2 and skewness angle = 53.5 degree. And the grid independence study was done.



Dimensions of the Conventional model



Dimensions of the new model

Fig. 1

Boundary conditions

As a first step the modeling of the air flow for the conventional single tangential air inlet geometry was performed and for this pressure swirl nozzle was considered in –z direction as shown in the figure 1.The outlet pressure is set to -100 Pa that is a fan is assumed to

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology
Vol. 2, Issue 6, June 2015

be draw the air out from the spray dryer . the velocity of the feed from the nozzle is set to the 59 m/s.
The nozzle was a pressure swirl nozzle (pre defined in the software) with the spray angle 76 degree. And rosin rammler parameter 2.05 (mean diameter of the particles 70.5µm).
And the detailed boundary conditions are in the Table 1 which used for the case study

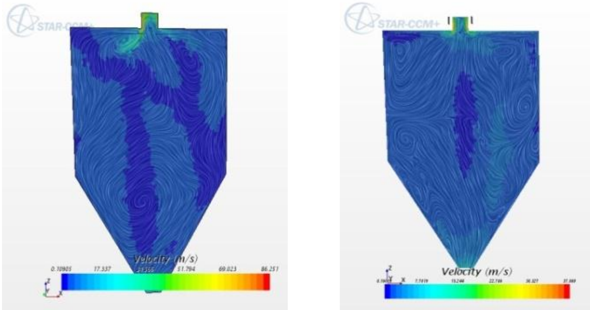
Table 1: Boundary conditions used for CFD analysis of spray dryer

Air mass flow rate (kg/s)	0.3863	0.19315/ inlet
Air temperature (K)	468 to 688	
Air absolute humidity (Kg of water/Kg of dry air)	0.014	
Spray rate (Kg/s)	0.01399	
Silica Content (% wt)	40	
Water Content in feed (slurry) (% wt)	60	
Initial moisture content of slurry ((Kg of water/Kg of dry silica)	1.5	
Pressure used to inject the slurry from nozzle (bar)	28.796	
The velocity of the particles at discharge from the nozzle (m/s)	59	
Spray Angle (degree)	76	
Rosin–Rammler parameter	2.05	
Feed temperature (K)	300	
Turbulence k-value (m^2/s^2)	0.027	
Turbulence ε-value (m^2/s^3)	0.37	
Pressure at outlet (Pa)	-100	
Chamber wall thickness (m)	0.002	
Wall material	Steel	
Wall-heat transfer coefficient (W/m^2 K)	3.5	
Air temperature outside wall (◦C)	27	
Interaction B.C. between wall and droplet	Escape	

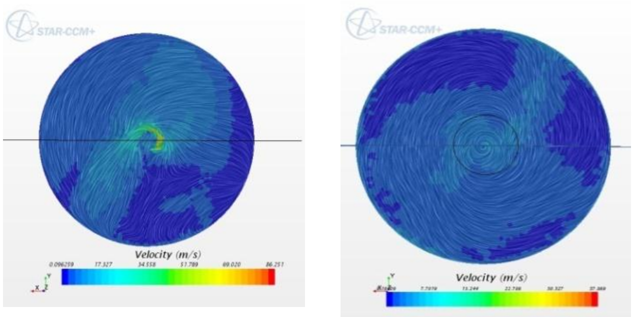
The same boundary conditions were used for the second modified balanced double tangential air inlet model. The mass flow rate of air is divided equally for the both inlets to keep the mass flow rate in both case same.
For the turbulence study k-ε was chosen because in most engineering practice it is the commonly used model, since k-ε model convergence considerably better than algebraic stress model and Reynolds stress model and it requires less computational efforts[6].
Chamber wall conditions: According to the options that could be selected for the present work in the STAR CCM+9.04 code, when a droplet/particle hits the wall of the drying chamber, it can be assumed to be "trapped" or "escaped" or "reflected" by the wall. In the CFD analysis of the co-current spray dryer chamber analysis the interaction between the surface (wall) of the spray dryer chamber and the particles injected into the spray dryer chamber is the "escape" type boundary condition is selected.
For the each particle trajectory, the location of the end point and the time of flight were recorded. In order to compare the models performance in both the case study.
The same boundary conditions are used to analysis work of both the case study

IV. RESULTS AND DISCUSSIONS

Conevtional spray dryer **Modified double air inlet spray dryer**

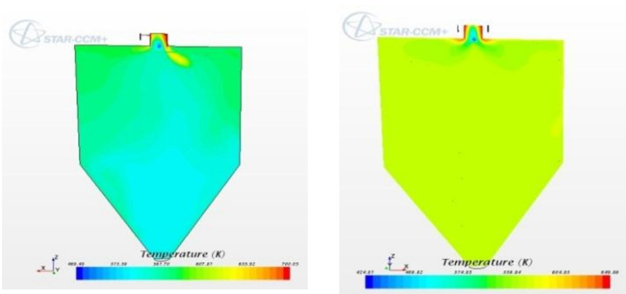


(a)



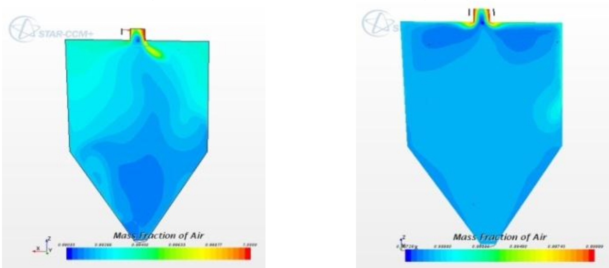
(b)

Fig.2 The contours of air velocity CFD analysis of the conventional spray dryer and the modified spray dryer



(c)

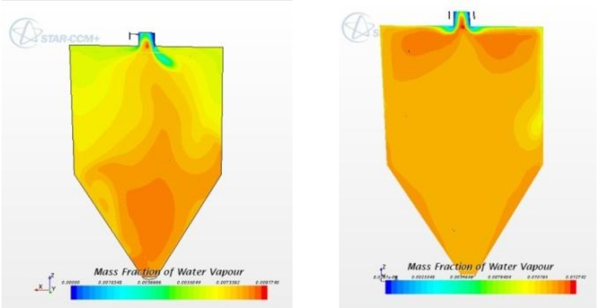
Fig.3 The temperature contours of the conventional spray dryer and the modified spray dryer



(d)

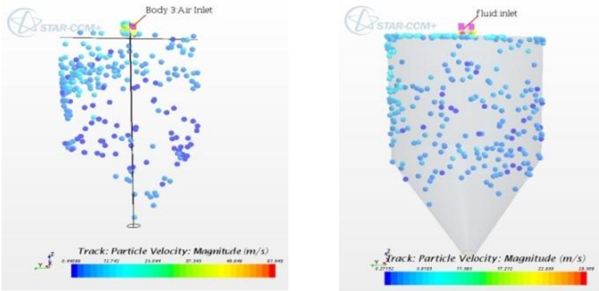
 DOI 10.17148/IARJSET.2015.2623

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
Vol. 2, Issue 6, June 2015



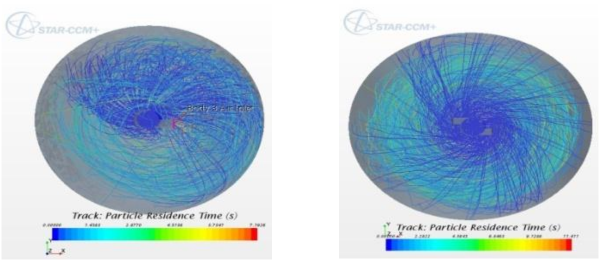
(e)

Fig.4 The mass fraction contours of CFD analysis of the conventional spray dryer and the modified spray dryer

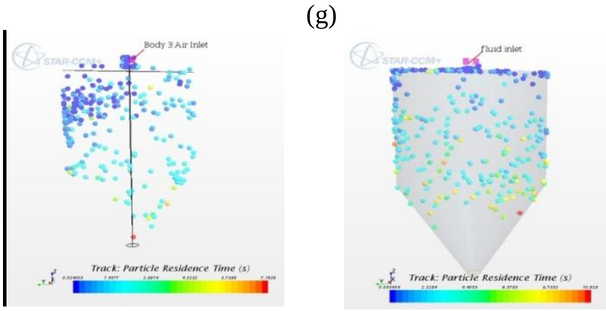


(f)

Fig.5 The contours particle velocity of CFD analysis of the conventional spray dryer and the modified spray dryer



(g)



(e)

Fig. The particle residence time contours of the conventional spray dryer and the modified spray dryer

Analysis of CFD simulation of air velocity profile with the comparison of both the case study as shown in the figure 2 (contours of air velocity). In this top view cross section and front view cross section were made for the analysis of the both the case study, which clearly describes the flow pattern in the conventional model is not a perfect swirl (shown in the top view contours) where as in the modified model the swirl is a complete and uniform over the plane. In the front view of the velocity contours the dark blue patch which represents the air velocity magnitude less then 0.1 m/s thus in this region particles velocity are more hence which hits the wall rapidly. So the wet particle with out drying stick to the wall of the spray dryer in the patched parts of conventional spray dryer and this problem is over came in the modified design of spray dryer. In the contours plots of temperature and mass fractions figure 3 and figure 4 it is clear that on the either side of the center axis of the spray dryer near the inlet, there is a imbalance in the flow occurs in the conventional spray dryer more flow occurs on the one side and less flow occurs on the other side, this is due to concentration gradient, on the inlet port side more concentration thus a greater band of temperature where as on the other side it is poor. But this problem is solved in the modified model by equal and uniform distribution on either side of the center axis of the spray dryer.

For the multiphase flow the particle track is done Lagrange approach in the software and got the results of contours as shown in the figure.5 In that the dark blue particles are of low velocity particles and light blue particles are of high velocity particles.

In the conventional model there is a perfect distinct of light and dark blue particles occurs in the chamber and these particles are non uniformly distributed on the conical part of the spray dryer and in the cylindrical part large amount of particles are accumulated on the left side when compared to right side.

But in the modified designed spray dryer model uniform distribution of the particle and with a uniform velocity through out the distributed particle in the chamber.

When the path of the particles tracked with respect to the residence time a clear picture of swirl is obtained from the top view as shown in the figure 6.

In the conventional model the swirl occurs only on one side at the top of the spray dryer chamber and swirl propagates to the down word. Where as in the modified model the swirl is complete and uniform over the each horizontal plane

And when the particle residence time tracked with the Lagrange approach, the problem of wet particle accumulation in the conventional model on the wall clearly described by the dark blue particles(residence time less than 0.1 second)which are on the left side. And highly dried particles distinct in colour on the other side which shown in the contour plots of track residence time.

The residence time of the particles in the modified design spray dryer no such less residence timed particles in the chamber thus the time foe the moisture to evaporate is sufficient in the new model for the particle before to hit the wall hence the problem of wet accumulation on wall is solved.

V. CONCLUSION

The internal flow behavior of the spray dryer is correctly predicted by the CFD simulations. The comparison of conventional model and the design modified model gives out the solution to the problems of the incomplete swirl and wet particles accumulation on the wall of spray dryer.

A uniform distributions of the particles in the spray dryer chamber occurs in the modified design spray dryer where as accumulation of particle on one side is in the conventional model.

Due to the complete swirl in the modified design model, the residence time of the particles has been increased for the same body structure of the spray dryer thus an effective heat and mass transfer occurs in this model.
Due to the uniform distribution of the temperature over the spray dryer, higher heat sensitive material can be used to dry using the modified design model and the out put dried product particles all are of at good specific properties.

ACKNOWLEDGEMENT

This project wouldn't have been successful without the help of many people. Many people contributed their knowledge and time to make this project successful one. I would like to extend my gratitude to all of them with the help of this acknowledgement. Without the effort of all these people I would not have completed this project successfully.
I would like to thank **Mr. Raghavendra N** Asst. Professor, Department , Mechanical Engineering, The oxford college of engineering, for his continuous support and guidance, I am also grateful to **Dr. N K S Rajan** , Chief Research Scientist, CGPL, IISC Bangalore for providing the necessary facilities to carry out the project.
I am also thankful to our **Dr. C. Badarinath**, Head of Mechanical Department and entire staff members for providing help and support.
I am grateful to **Dr. Rajendra Prasad** our beloved principal for his great care and custody bestowed on me.
My special thanks go to **Mr. Pranay kale** project assistants for their moral support and constant help during the project.

REFERENCES

1. Masters, K., Spray Drying Handbook, 4^{th} ed., John Wiley and Sons, New York,1985.
2. Oakley, D., "Produce uniform particles by spray drying", Chem. Eng. Prog., October, pp. 48-54, 1997.
3. Harvie, D.J.E., Langrish, T.A.G and Fletcher D.F., "A computational fluid dynamics study of a tall-form spray dryer". Trans. IChemE C, 80(3), 163-175, 2002.
4. Harvie, D.J.E.; Langrish, T.A.G.; Fletcher, D.F., "Numerical simulations of gas flow patterns within a tall-form spray dryer". Transactions of the Institution of Chemical Engineers,79 (3) A, 235 248, 2001.
5. Oakley, D.E.; Bahu, R.E.,"Computational modeling of spray drying,Comput". Chem. Eng. 17 (1993) 493-498.
6. Kieviet, F.G. Modelling Quality in Spray Drying. Ph.D. thesis, Endinhoven University of Technology, the Netherland, 1997.
7. Crowe, C.T., Sharma, M.P. , Stock, D.E., " The particle-source-in-cell (PSI-Cell) model for gas-droplet flows", Journal of Fluid Engineering, 99, 325 332, 1977.
8. User Guide STAR CCM+9.04, CD-adpco.